2D Consolidation under a Strip Footing
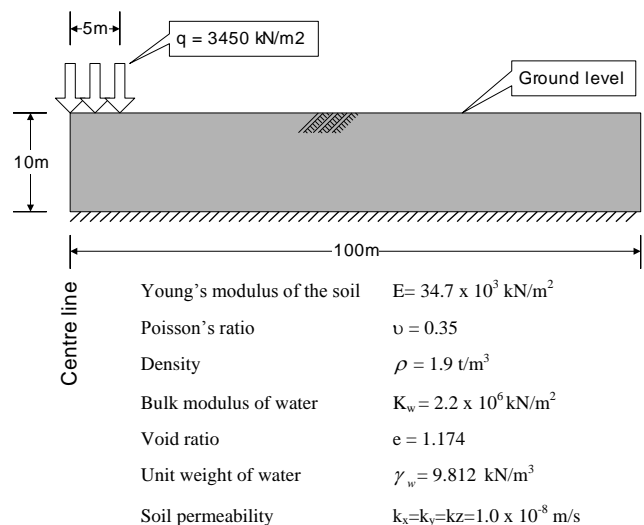
For LUSAS version:	18.0
For software product(s):	Any Plus version.
With product option(s):	Nonlinear, Dynamic.
Note: The example exceeds the limits of the LUSAS Teaching and Training Version.	

Description

The pore water pressure dissipation and settlement in a soil following the application of a distributed load is to be investigated.

Horizontal displacement at the left edge is restrained to model the symmetrical boundary condition. Vertical displacement at the base of the soil is restrained as the soil rests on solid rock. The pore pressure at ground level is prescribed to zero.

Units of kN, m, t, s, C are used throughout.

Young's modulus of the soil	E= 34.7 x 10^3 kN/m^2
Poisson's ratio	$\upsilon = 0.35$
Density	$\rho = 1.9$ t/m^3
Bulk modulus of water	$K_w = 2.2$ x 10^6 kN/m^2
Void ratio	e = 1.174
Unit weight of water	$\gamma_w = 9.812$ kN/m^3
Soil permeability	$k_x = k_y = k_z = 1.0$ x 10^{-8} m/s

Objectives

The required output from the analysis consists of:

- ❑ Settlement at the centre of the footing with time.
- ❑ Pore pressure distribution immediately after application of the load (undrained response)
- ❑ Pore water pressure dissipation with time at the centre of the footing

Keywords

2D, Plane Strain, Consolidation, Pore Water Pressure, Time Stepping, Nonlinear, Transient, Settlement, Two Phase Materials, Default Assignments, Graphs

Associated Files

- ❑ **pwp_modelling.vbs** carries out the modelling of the example.

Modelling

Running LUSAS Modeller

For details of how to run LUSAS Modeller, see the heading *Running LUSAS Modeller* in the *Introduction to LUSAS Worked Examples* document.

Note. This example is written assuming a new LUSAS Modeller session has been started. If continuing from an existing Modeller session select the menu command **File>New** to start a new model file. Modeller will prompt for any unsaved data and display the New Model dialog.

Creating a new model

- Enter a File name of **pwp**

- Use the default User-defined working folder.

- Ensure an Analysis type of **Structural** is set.

- Select an Analysis Category of **2D Inplane**

- Set Model units of **kN,m,t,s,C**

- Leave the Timescale units as **Seconds**

- Select a Startup template of **2D Plane Stress**.

- Ensure the Layout grid is set as **None**

- Enter a Title of **Consolidation under a Strip Footing**

- Click the **OK** button.

✎ **Note.** The Undo button may be used to correct a mistake. The undo button allows any number of actions since the last save to be undone.

Mesh Definition

Since this analysis requires the modelling of pore water pressure, plane-strain two phase elements will be used.

Attributes	
Mesh	>
Surface...	
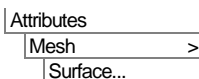
- Select **Plane strain two phase, Quadrilateral, Quadratic** elements. Ensure the **Regular mesh** option is selected with **Automatic** chosen so that Modeller will use the mesh divisions assigned to each line.

- Give the attribute the name **Plane Strain Two Phase** and click the **OK** button to add the mesh attribute to the 🔬 Treeview.

- In the 🔬 Treeview click on the mesh attribute **Plane Strain Two Phase** with the right-hand mouse button and select the **Set Default** option**.** This will ensure all newly created surfaces will be assigned the elements defined in this mesh attribute.

Material Properties

For consolidation analysis both the elastic and two-phase soil properties need to be defined. The overall 'equivalent' bulk modulus of the soil is related to the bulk modulus of the pore fluid and the bulk modulus of the solid soil particles by the formula:

$$\frac{1}{Ke} = \frac{n}{Kf} + \frac{(1-n)}{Ks} \approx \frac{n}{Kf}$$

Where

Ke is the equivalent bulk modulus of the soil

Kf is the bulk modulus of the pore fluid

Ks is the bulk modulus of the solid soil particles

n is the porosity of the soil

The porosity of the soil is related to the void ratio by the formula:

$$\text{Porosity } n = \frac{e}{1+e} = 0.54$$

- Enter the isotropic material properties for the soil as Young's Modulus **34.7E3**, Poisson's Ratio **0.35** and Mass density **1.9**

- Click on the **Two phase** check box at the top right-hand corner of the dialog.

Isotropic ✕

☐ Plastic   ☐ Creep   ☐ Damage   ☐ Shrinkage   ☐ Viscous   ☑ Two phase   ☐ Ko Initialisation

Elastic | Two Phase

⦿ Fully saturated

○ Partially saturated

○ Piecewise linear partially saturated

Define curves...

☐ Incompressible solids

	Value
Bulk modulus of solid phase	2.2e6
Bulk modulus of fluid phase	2.2e6
Porosity of medium	0.54
Hydraulic conductivity in global X direction	0.01e-6
Hydraulic conductivity in global Y direction	0.01e-6
Hydraulic conductivity in global Z direction	0.01e-6
Density of fluid	0.1

Name Fissured Clay ▼ ⯅⯆ (new)

OK   Cancel   Apply   Help

- For a **Fully saturated condition** enter the properties as shown in the dialog above.

Note. For the purposes of this example and to comply with a corresponding test case the undrained bulk modulus of the soil is set to be the same as that of the pore fluid.

- Enter the attribute name as **Fissured Clay** and click the **OK** button to add the attribute to the 🔗 Treeview.

- In the 🔗 Treeview click on the attribute **Fissured Clay** with the right-hand mouse button and select the **Set Default** option. This will ensure all newly created features will be assigned the properties defined in this material attribute.

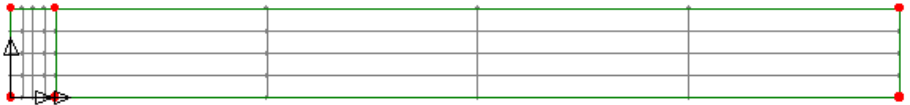## Feature Geometry

Geometry
Surface   >
  By Coords...

 Enter coordinates of (**0, 0**), (**5, 0**), (**5, 10**) and (**0, 10**) to define the soil under the load. Use the Tab key to move to the next entry field on the dialog. When all coordinates have been entered click the **OK** button.

Geometry
Surface   >
  By Coords...

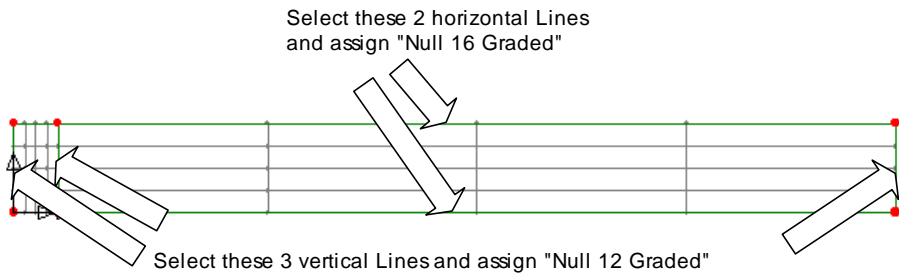 Enter the coordinates of (**5, 0**), (**100, 0**), (**100, 10**), (**5, 10**) to define the remainder of the soil.

Mesh Grading

Attributes
Mesh   >
  Line...

- Define a Line mesh with element type **None** and **16** divisions.

- Select the **Spacing** button.

- Choose the **Uniform transition** option and set the Ratio of first to last element to **0.1**

- Click **OK** to accept the spacing properties.

- Give the attribute name as **Null 16 Graded** and click the **Apply** button to add the attribute to the Treeview and keep the dialog active.

- Define a Line mesh with **12** divisions.

- Select the **Spacing** button.

- Enter the **Uniform transition** option and set the Ratio of first to last element to **5**

- Click **OK** to accept the spacing properties.

- Give the attribute name as **Null 12 Graded** and click **OK** to add the attribute to the Treeview.

Select these 2 horizontal Lines
and assign "Null 16 Graded"

Select these 3 vertical Lines and assign "Null 12 Graded"

- Select the horizontal lines of the larger surface as shown and assign the mesh attribute **Null 16 Graded**

- Select all the vertical lines as shown (Use the **Shift** key to add to the initial selection) and assign the mesh attribute **Null 12 Graded**

Note. If the mesh is graded with the smaller elements at the wrong end of a line reverse the line by selecting the line, and using the **Geometry>Line>Reverse** menu. The end result should be a mesh that looks like this:
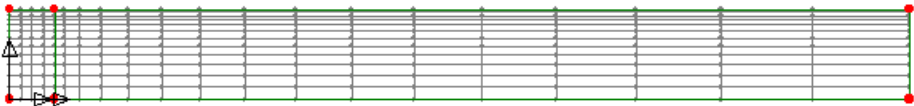
Supports

Assign the supports to the model.

- Select the vertical Line on the left-hand side of the model and drag and drop the support attribute **Fixed in X** from the 🔅 Treeview onto the selection. With the **Assign to lines** option selected click **OK** to assign the support attribute to **All analysis loadcases**

- Select the 2 horizontal Lines representing the base of the soil and drag and drop the support attribute **Fixed in Y** from the 🔅 Treeview onto the selection. With the **Assign to lines** option selected click **OK** to assign the support attribute to **All analysis loadcases**

Loading

To define the distributed load:

Attributes
Loading...

- Select the **Face** option and click **Next**

- Enter loading of **3450** in the **y Direction**

- Enter the attribute name of **Distributed Load** and click the **Finish** button.

- Select the top line on the left-hand side of the model and drag and drop the **Distributed Load** attribute onto the selection. With the **Assign to lines** option selected click **OK** to assign the loadcase attribute to **All loadcases**.

Select this line
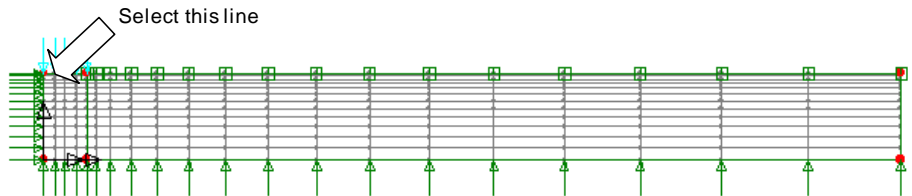
Analysis Control

Undrained loading stage

The undrained loading of the soil must be calculated to establish the initial conditions of the consolidation analysis.

- In the Treeview expand **Analysis 1** then right-click on **Loadcase 1** and select **Nonlinear & Transient** from the **Controls** menu.

In the Incrementation section:

- Select the **Nonlinear** option and choose **Manual** incrementation.

- Select the **Time domain** option and chose **Consolidation** from the drop down list.

- Enter an **Initial time step** of **1**

- Enter **Total Response Time** of **1**.

- Set **Max time steps or increments** to **0.**

Consolidation stage

To allow dissipation of the excess pore-water pressures a drainage path must be defined which for this example is through the ground level of the soil. A new loadcase needs to be created for this.

Analyses
Loadcase…

- Click **OK** to create **Loadcase 2**

Supports

Define a support to prescribe zero pore pressure.

Attributes
Support…

- Specify the **Pore pressure** to be **Fixed.**

- Enter the attribute name as **Drainage Path** and click the **OK** button.

- Select the Line at ground level on the right-hand side of the model (see image) and drag and drop the support attribute **Drainage Path** from the 🔩 Treeview onto the selection. With the **Assign to lines** option selected, choose the **From loadcase (nonlinear and transient) option** to assign the supports to **Analysis 1** and **Loadcase 2**.Click **OK**.

Select this line

Loading
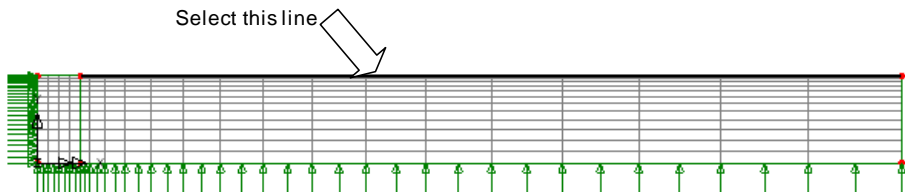
Attributes
Loading...

- Select the **Prescribed displacement** option and click **Next**

- With the **Total** option selected, set the Pore Pressure to be **Fixed** and enter a prescribed displacement of 0. Name the attribute **Surface Excess Pore Water Pressure** and click **OK**.

- Select the Line at ground level on the right-hand side of the model (see image above) and drag and drop the loading attribute **Surface Excess Pore Water Pressure** from the 🔾 Treeview onto the selection for a **Single loadcase** named **Loadcase 2**.

Analysis Control

Consolidation stage

With this consolidation problem an automatic time stepping procedure is adopted. This is because consolidation is a typical diffusion process in which the field changes rapidly at the start of the process before settling down to a steady state condition a considerable time after the initial load is applied. The automatic time stepping procedure enables the time step to be modified so that the small time steps required at the start of the problem can be increased as the analysis progresses. In some cases the overall response time can be orders of magnitude larger than the initial time step.

The initial time step is important since the early variations in pore pressure must be accurately accounted for. Vermeer and Verruijt suggest the following criteria for determining the initial time step where Δh is the minimum distance between nodes.

$$\Delta t \geq \frac{\gamma_\omega}{6\mathrm{E}k}(\Delta h)^2 = \frac{9.812}{6 \text{ x } 34.7E3 \text{ x } 1E-8} \text{ x } 0.14^2 \approx 100 \text{ secs}$$

🖉 **Note.** The distance between two nodes can be determined by selecting the two nodes and then picking the **Tools> Mesh> Distance between Nodes** menu item.

- In the 🕘 Treeview expand **Analysis 1** then right-click on **Loadcase 2** and select **Nonlinear & Transient** from the **Controls** menu.

In the Incrementation section:

- Select the **Nonlinear** option and choose **Manual** incrementation.

- Select the **Time domain** option and chose **Consolidation** from the drop down list.

- Enter an **Initial time step** of **100**

- Enter **Total Response Time** of **100E6**.

- Set **Max time steps or increments** to 5**0.**

- Select the **Advanced** button in the Time domain section of the dialog.

- On the Advanced time step parameters dialog set the 'Time step increment restriction factor' to **2**

- Set the 'Minimum time step' to **100**

- Ensure the 'Maximum time step' is set to **100E6**

- Ensure the 'Minimum time step for termination' is set to **0**

- Click **OK** to return to the Nonlinear and Transient control dialog.

- Click **OK** to set the loadcase control.
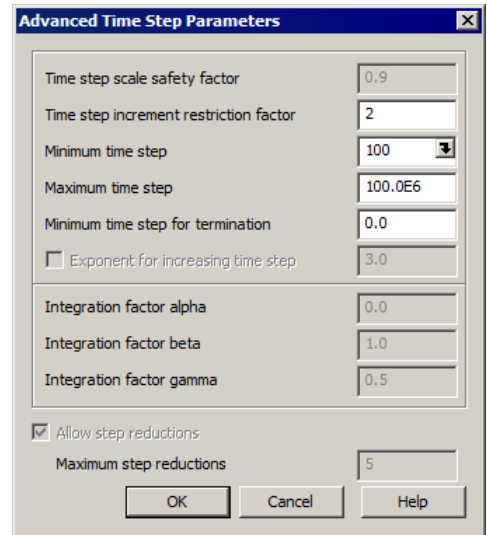
Saving the model

File
Save

Save the model file.

Running the Analysis

Open the **Solve Now** dialog. Ensure A**nalysis 1** is selected and press **OK** to run the analysis.

A LUSAS Datafile will be created from the model information. The LUSAS Solver uses this datafile to perform the analysis.

If the analysis is successful…

Analysis loadcase results are added to the ⏰ Treeview.

In addition, 2 files will be created in the Associated Model Data directory where the model file resides:

❑ **pwp.out** this output file contains details of model data, assigned attributes and selected statistics of the analysis.

❑ **pwp.mys** this is the LUSAS results file which is loaded automatically into the Treeview to allow results processing to take place.

If the analysis fails...

If the analysis fails, the output file will provide information relating to the nature of the error encountered. Any errors listed in the output file should be fixed in LUSAS Modeller before saving the model and re-running the analysis.

Rebuilding a Model

If it proves impossible for you to correct the errors reported a file is provided to enable you to re-create the model from scratch and run an analysis successfully.
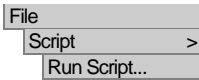
❑ **pwp_modelling.vbs** carries out the modelling of the example.
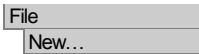
File
New…

Start a new model file. If an existing model is open, Modeller will prompt for unsaved data to be saved before opening the new file.

- Enter the file name as **pwp**

File
Script >
Run Script...

- To recreate the model, select the file **pwp_modelling.vbs** located in the **\<LUSAS Installation Folder>\Examples\Modeller** directory.

Rerun the analysis to generate the results.

Viewing the Results

Loadcase results can be seen in the Treeview. For a nonlinear analysis the last solver results time step is set active by default.

- If present, turn off the **Geometry**, **Mesh** and **Attributes** layers in the Treeview.
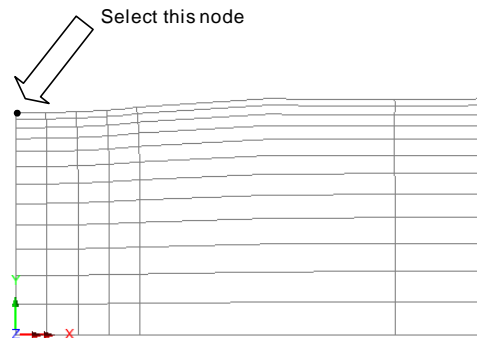
Settlement

- In the Treeview, right-click on the first load increment **Time Step 0** and select the **Set Active** option to view results immediately after the loading is applied.

- Click the **Deformations** button in the Treeview and select the **Specify factor** option. Specify a factor of **1** and click **OK** to visualise the deformed mesh for the first time step.
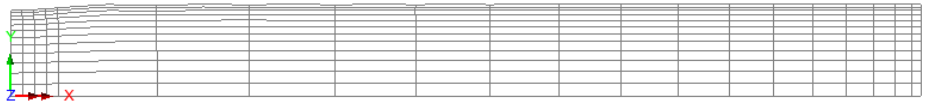
A graph of the deformation over time will be created using the graph wizard.

- Zoom into the left-hand side of the model and select the node on the centre line at the centre of the footing.

Select this node

Graphing settlement

- With the **Time history** option selected click **Next**

Firstly define the data to be used for the X axis.

- Selected the **Named** option and click **Next**

- Select **Response Time** from the drop down list and click **Next**

Secondly define the data for the Y axis.

- With the **Nodal** option selected and click **Next**

- Select **Displacement** from the Entity drop down list and **DY** from the Component drop down list.

- The selected node number will appear in the **Specified single node** drop down list. Click **Next**
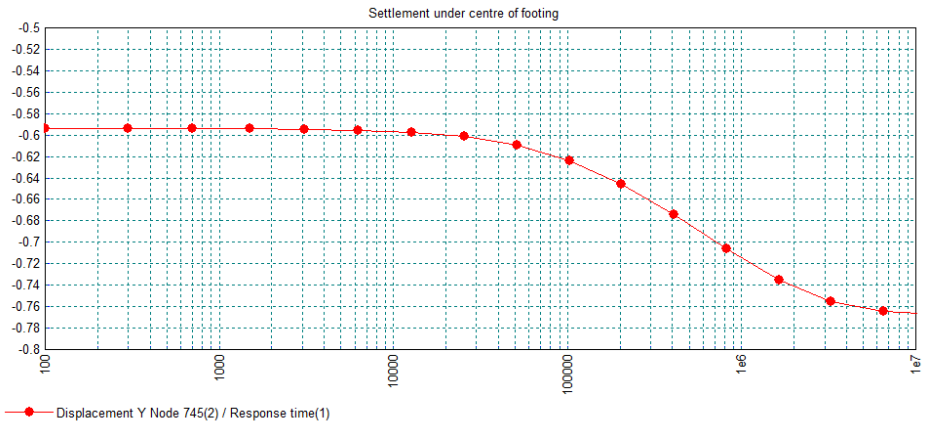
For the X Scale enter **Manual** values of minimum **100** and maximum **10e6** respectively.

- Select the **Use logarithmic scale** option.

- Title the graph as **Settlement under centre of footing**

- Click **Finish** to display the graph of settlement over time under the centre of the strip footing.

Settlement under centre of footing

— Displacement Y Node 745(2) / Response time(1)

☒ Close the graph window

Pore water pressure

The distribution of pore pressure is to be shown using contours.

- With **Time Step 0** set active in the ⌖ Treeview, and with no features selected, click the right-hand mouse button in a blank part of the view window and select **Contours** to add the contours layer to the ⬓ Treeview.

- Select **Displacement** from the Entity drop down list and **PRES** from the component drop down list.

- Select the **Contour Range** tab and set the **Interval** to **250**

- Click the **OK** button to display contours of the undrained pore pressure distribution (Time step 0) immediately after the loading is applied.

Analysis: Analysis 1
Loadcase: 1:Loadcase 1, 1:Time Step 0 Time = 0.000000E+00
Results file: pwp~Analysis 1.mys
Entity: Displacement
Component: PRES (Units: kN/m²)

-750.0
-500.0
-250.0
0.0
250.0
500.0
750.0
1.0E3
1.25E3
1.5E3
1.75E3
2.0E3
2.25E3
2.5E3
2.75E3
3.0E3

Maximum 3.14978E3 at node 751
Minimum -979.176 at node 755
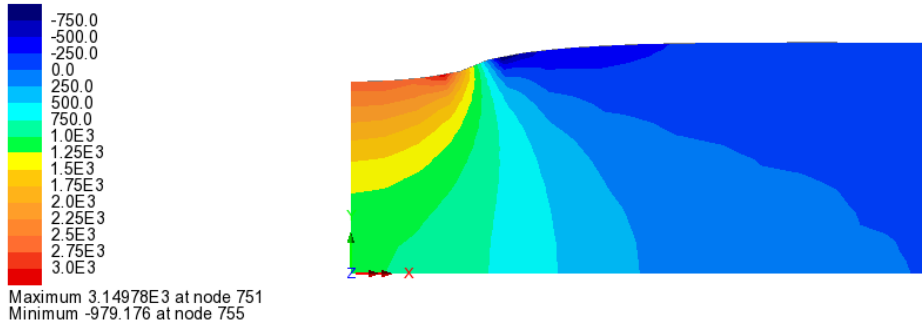
Note. The dissipation of pore water pressure over time may be observed by creating an animation of the contour display.

- To observe the distribution of pore water pressure at a particular time after the application of loading, activate the appropriate time step from the ⏱ Treeview by selecting the time step with the right-hand mouse button and choosing the **Set Active** option.

Graphing pore water pressure

The dissipation of pore water pressure under the footing is to be presented on a graph.

- Select the node on the centreline under the footing as before.

| Utilities |
| Graph Wizard... |

- With the **Time history** option selected click **Next**

Firstly define the data to be used for the X axis:

- Selected the **Named** option and click **Next**

- Select **Response Time** from the drop down list and click **Next**

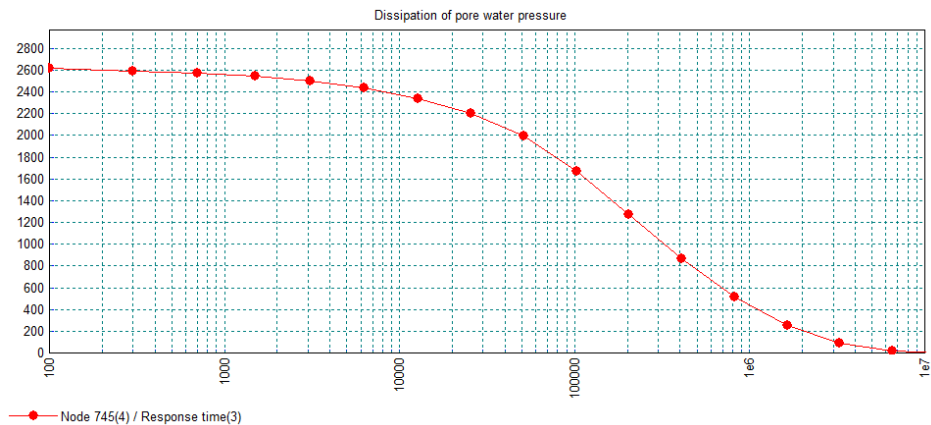Secondly define the data for the Y axis:

- With the **Nodal** option selected and click **Next**

- Select **Displacement** from the Entity drop down list and **PRES** from the Component drop down list.

- The extent to be graphed will be set to **Specified single node** and the node selected will be seen in the Specify node field. Click **Next**

- For the X Scale ensure that **Manual** values of minimum **100** and maximum **10e6** respectively are used.

- Ensure the **Use Logarithmic scale** option is selected.

- For the Y Scale leave the values as the defaults.

- Title the graph as **Dissipation of pore water pressure**

- Click **Finish** to display the dissipation of pore water pressure over time under the centre of the strip footing.

Dissipation of pore water pressure

—●— Node 745(4) / Response time(3)

[×] Close the graph window.

Graphing effective stress in soil

As the pore water dissipates the load is carried by the soil. The increase in effective stress in soil can be observed on a graph of effective stress against response time.

- Ensure the node on the centreline under the footing still selected.

| Utilities |
| Graph Wizard... |

- With the **Time history** option selected click **Next**

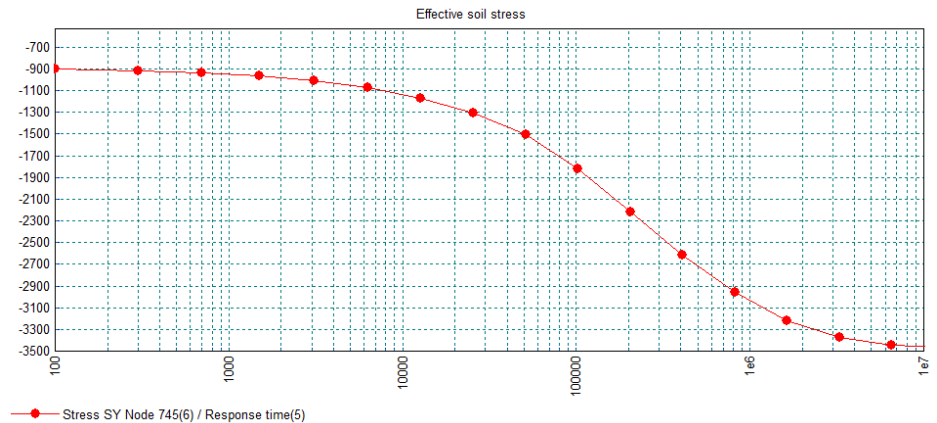Firstly define the data to be used for the X axis

- Select the **Named** option and click **Next**

- Select **Response Time** from the drop down list and click **Next**

Secondly define the data for the Y axis

- With the **Nodal** option selected and click **Next**

- Select **Stress** from the Entity drop down list and **SY** from the Component drop down list.

- The extent to be graphed will be set to **Specified single node** and the node selected will be seen in the Specify node field. Click **Next**

- For the X Scale ensure that **Manual** values of minimum **100** and maximum **10e6** respectively are used.

- Ensure the **Use Logarithmic scale** option is selected.

- For the Y Scale leave the values as the defaults.

- Title the graph as **Effective soil stress**

- Click **Finish** to display the graph of the effective soil stress over time under the centre of the strip footing.

Effective soil stress

Stress SY Node 745(6) / Response time(5)

This completes the example.